L8: ALTIUM BOARD LAYOUT I

ESE516: IoT Edge Computing

Monday, February 18, 2019

Eduardo Garcia - edgarc@seas.upenn.edu

ADMINISTRATIVE

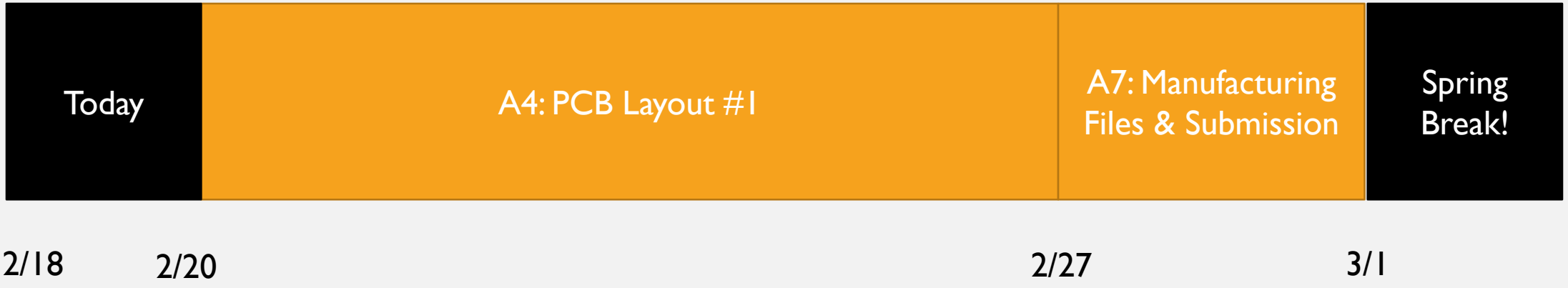
ADMINISTRATIVE

1. Any long lead-time items?
    1. I will place order for SAMW25 this week – low supply on U.S.
    2. If you require any components with long lead time please let me know
    3. People that need the HX711 – see me after class

2. I will stay after class to help with schematic reviews

3. If you have run into any issue please let us know as soon as possible so we may help!

SCHEDULE UNTIL SPRING BREAK



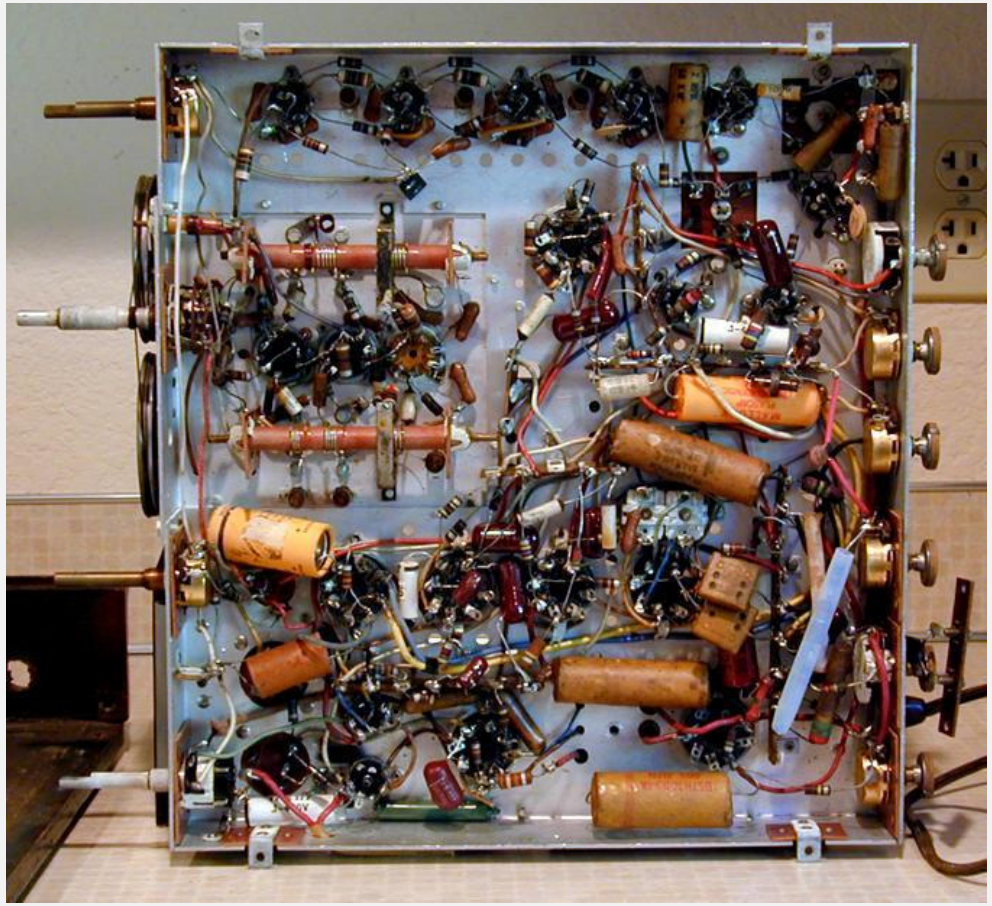
INTRO TO PRINTED CIRCUIT BOARDS

LECTURE GOALS

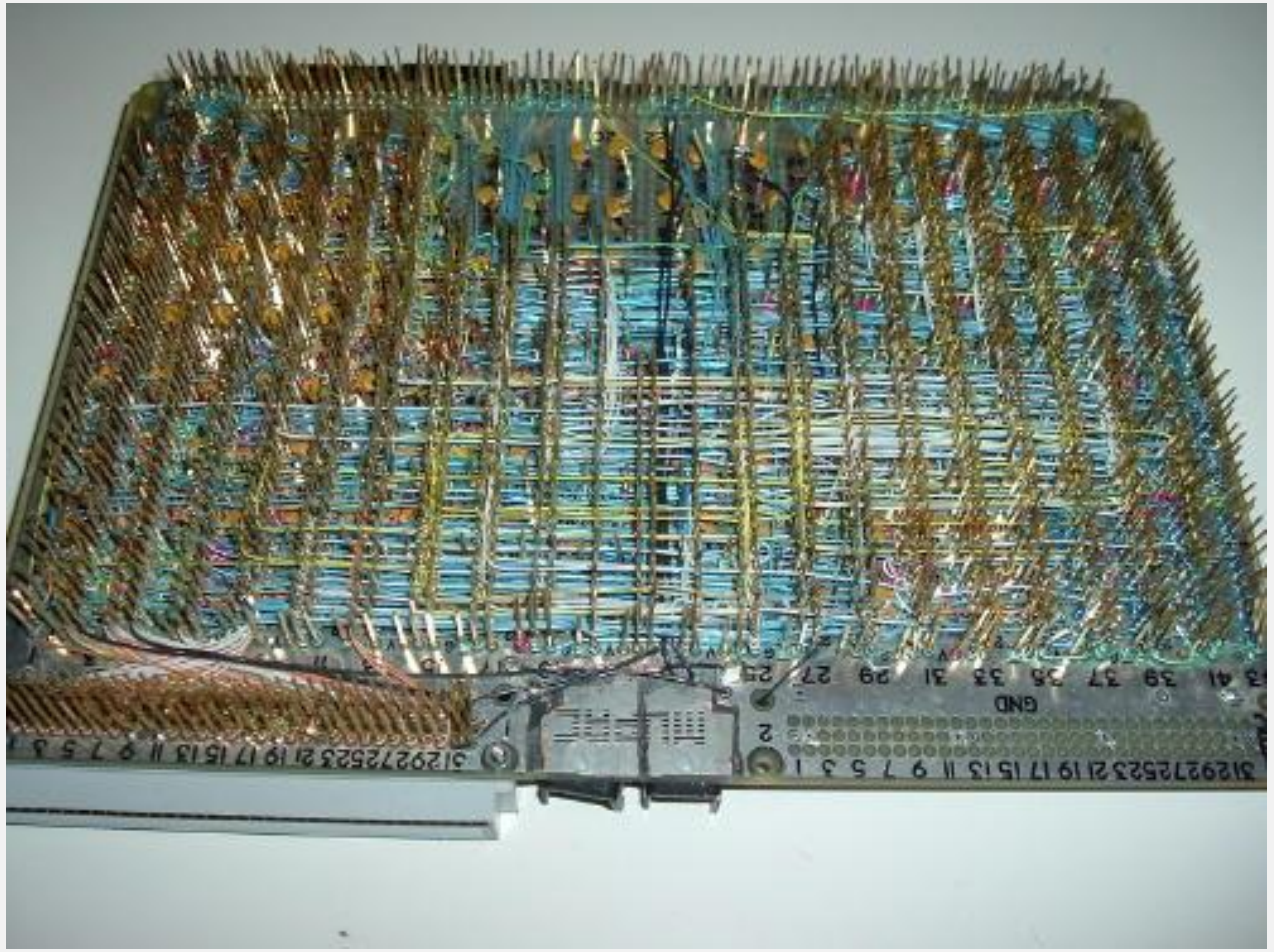
1. Introduction to PCB – Printed Circuit Board

2. Time Permits – Schematic Review

BEFORE PCBS



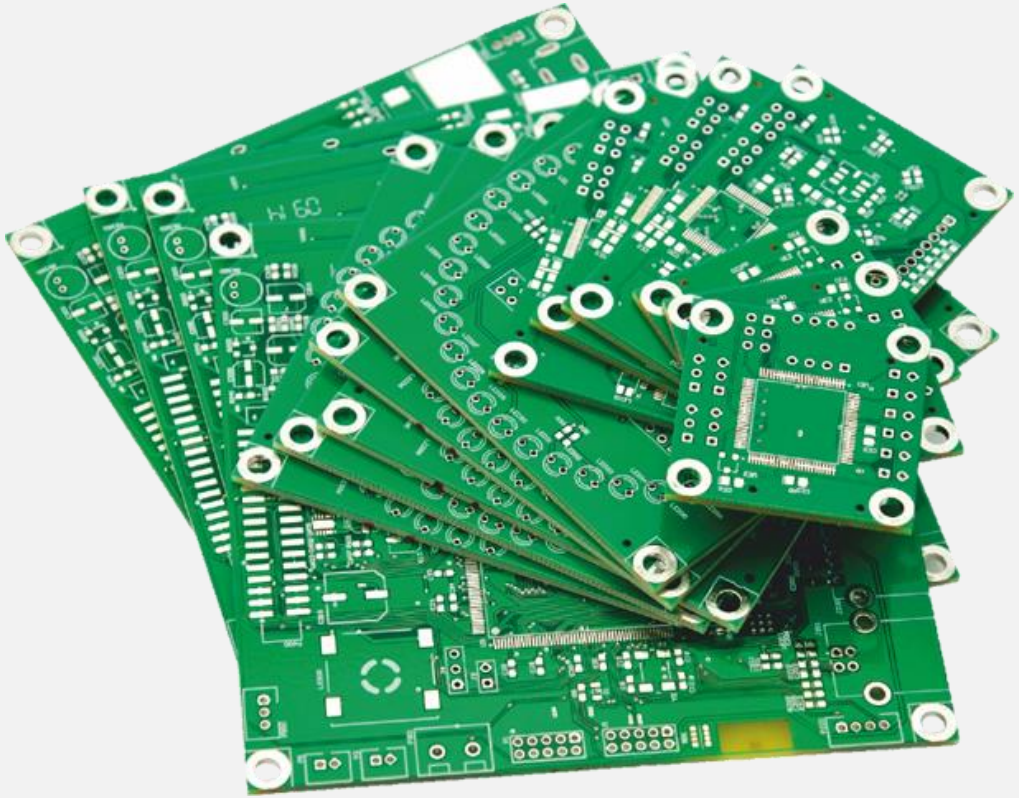
- Wire-Wound



WHAT IS A PCB

- **PCB: Printed Circuit Board**
- "*A PCB (Printed Circuit Board) is a board used to electrically connect and mechanically support components using pads, conductive tracks and several other features etched from sheets of copper laminated onto non-conductive substrates*"

- Taken from "https://en.wikipedia.org/wiki/Printed_circuit_board"



1. Taken from https://www.transonics.com/pcb-services

WHAT IS A PCBA (OR PCA)

- **PCBA: Printed Circuit Board Assembly**
- A PCBA board assembly refers to the completely assembled PCB with the components added and soldered into the board.

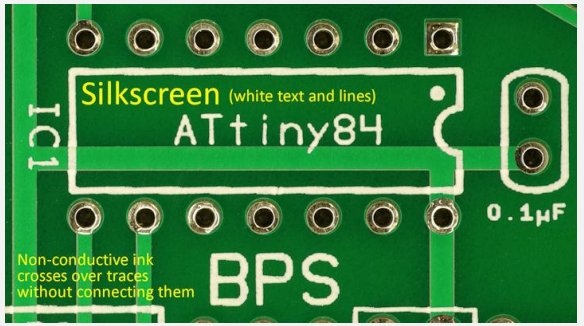
1. Taken from https://www.tme.eu/en/details/atatmel-ice-pcba/programmers/microchip-atmel/

PARTS OF A PCB (OVERVIEW)

- **PCBs** are fabricated by doing layers of different material on top of another.
- **Copper layers** are separated one from another by a **dielectric material.**
  - Copper thickness is measured usually by ounces and is the thickness of 1 ounce of **copper** rolled out to an area of 1 square foot.
- The top and bottom of the PCB have a layer called **Solder Mask** that prevents the copper from being exposed to air (except when we want to – i.e.: Component pads)
- An **Overlay** is added to bottom and top faces for indication.

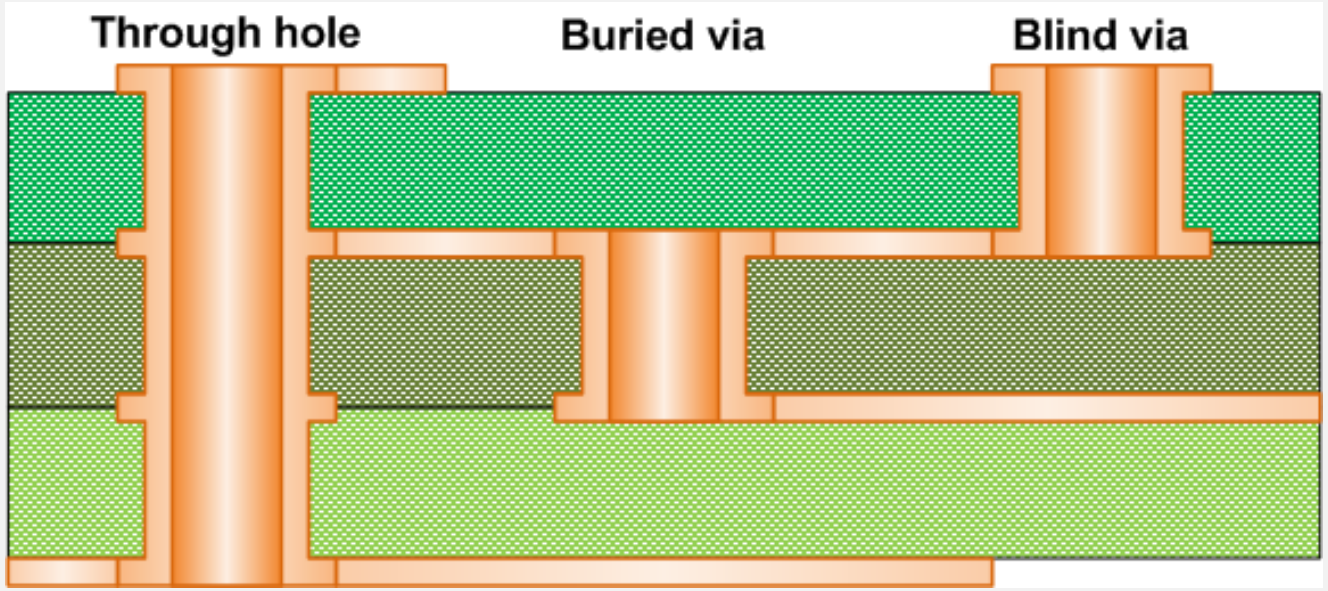
2 Layers **1.6mm (0.062 inch) Green for 1 Oz**

Layer stack up	Layer	Thickness (mm)
	Top Solder Mask	0.01
	Top Layer	0.035
	Core	1.5
	Bottom Layer	0.035
	Bottom Solder Mask	0.01



PARTS OF A PCB (OVERVIEW)

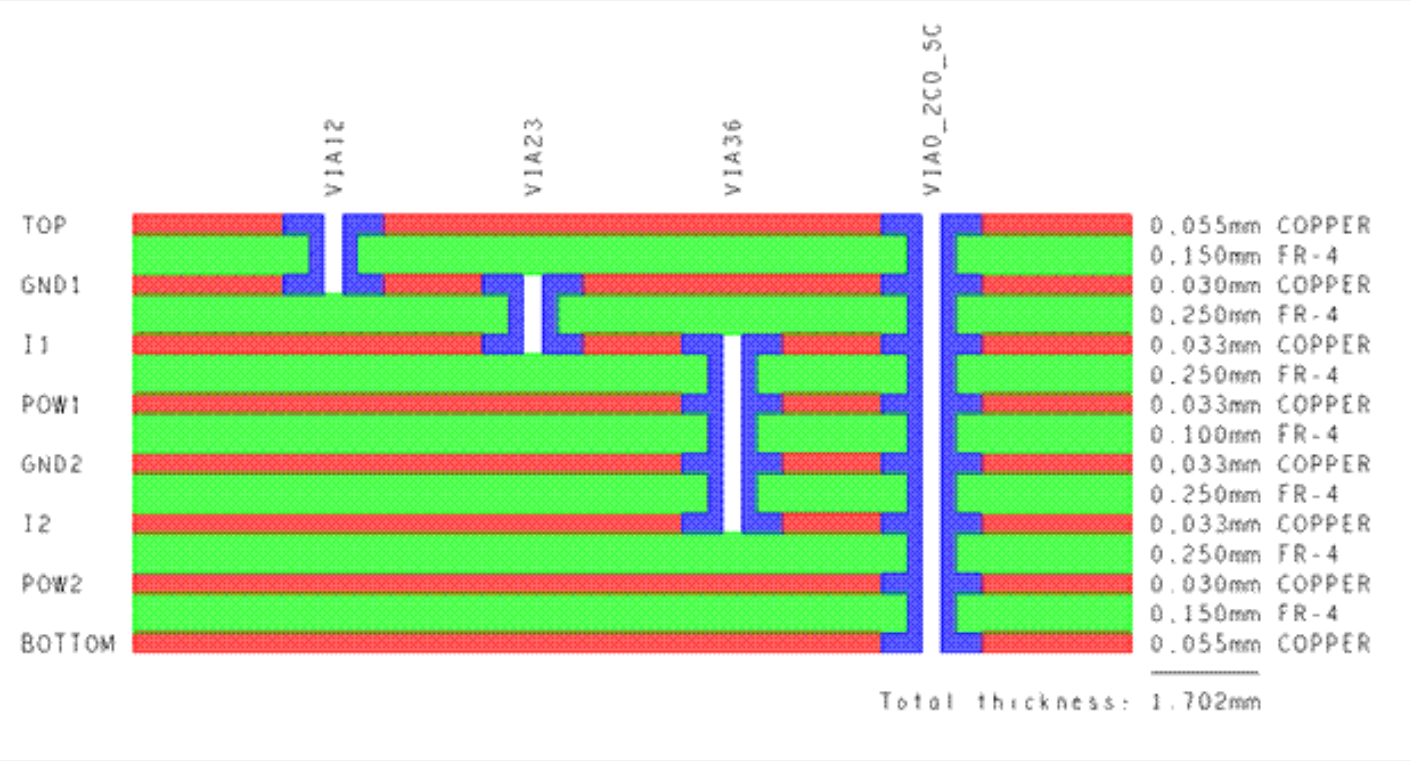
- Copper planes can be interconnected with the use of **vias**.
- Different types of vias:
- **Through-Hole vias:** Go through all the board. The easiest and cheapest to fabricate.
- **Blind Via** and **Buried vias** only connect to some of the layers. They increase the price and difficulty of fabrication.

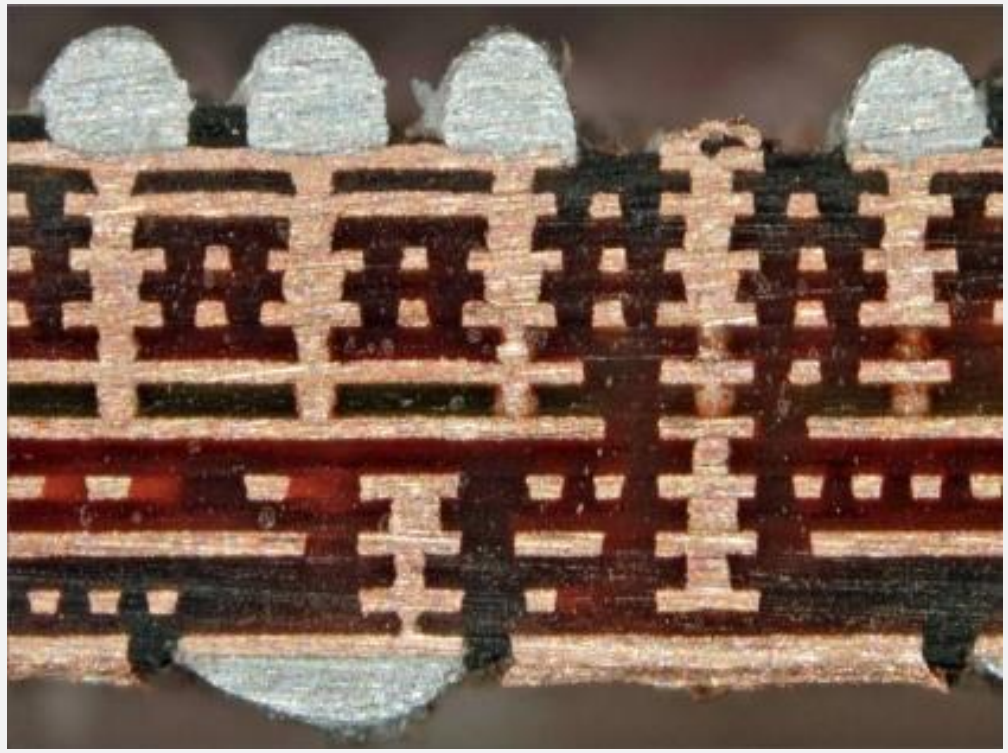


PCB STACKUP

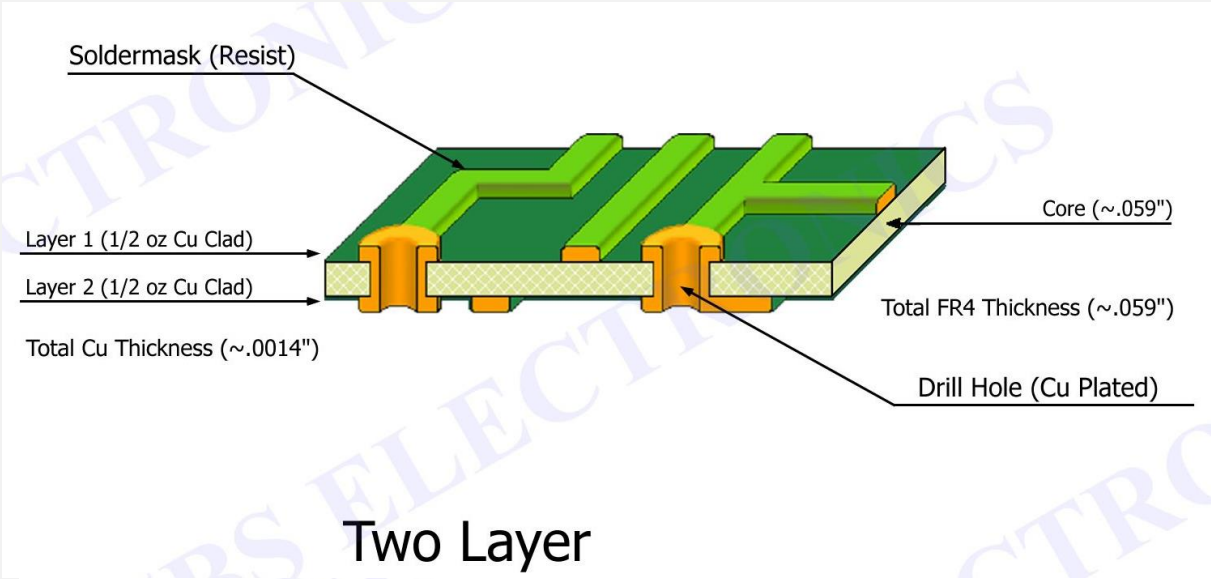
- The PCB Stackup describes the layers of a PCB and details how they are to be manufactured. They also describe if the copper layer is to be mainly for signals or for power (mixed use is allowed with some considerations).

- Manufacturers offer information on their stack-up capabilities – check with them to see what they can offer you!

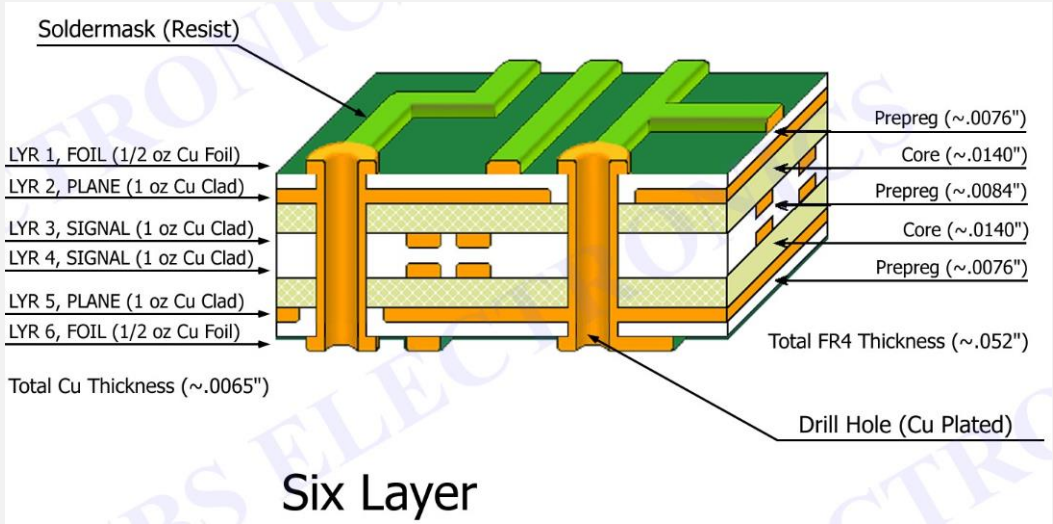




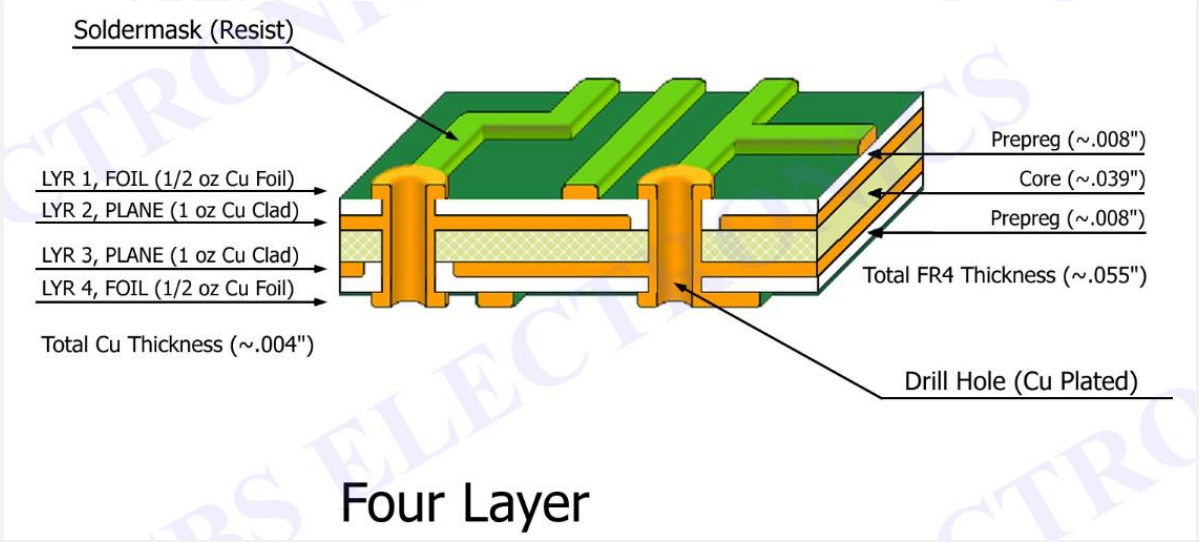
COMMON PCB STACKUPS



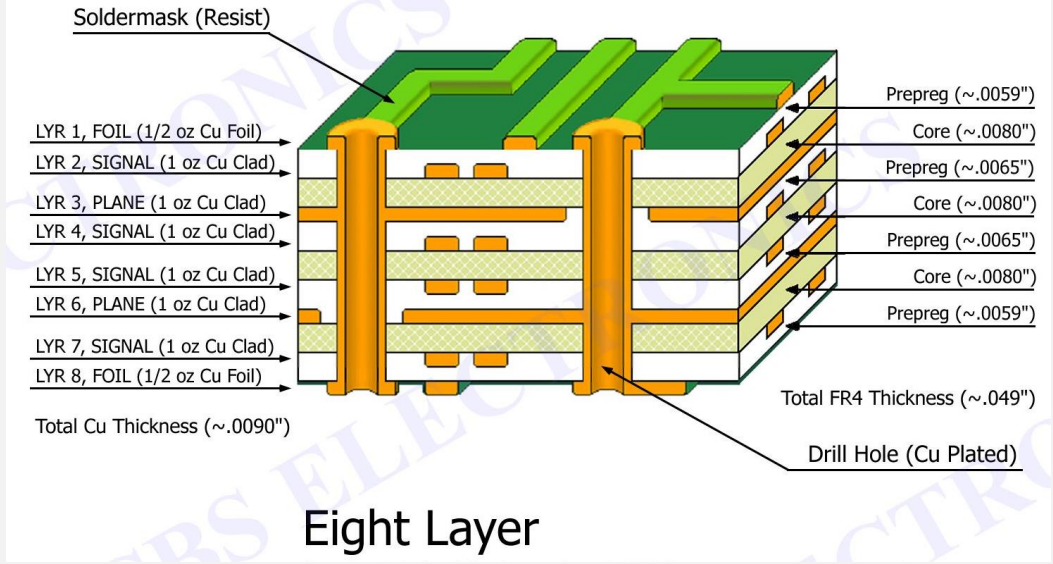
Two Layer



Six Layer



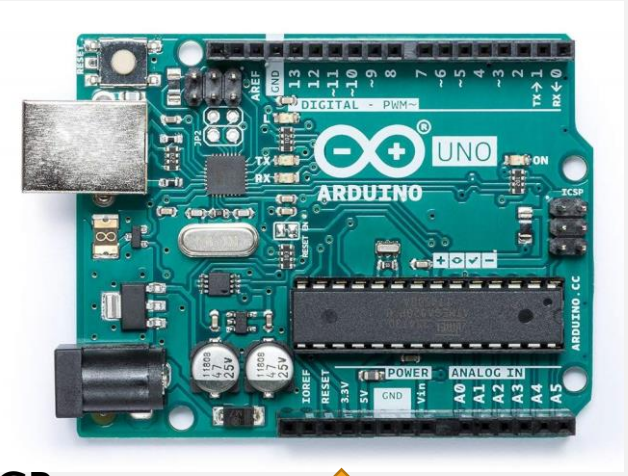
Four Layer



Eight Layer

PCB ARE A MANUFACTURING CHALLENGE

- Designing PCBs must be done with manufacturability in mind!
- It is not enough to make a PCB and then "throw it" to the manufacturer – the manufacturer has to be able to build it!
- It is not a good excuse to say "But how do cellphones do it?". Inside the PCB manufacturing realm there are different techniques at different costs.
- **Good Practice:** Talk early with your manufacturer, and understand their capabilities and limitations. Then design based on those limitations.
- **DO NOT:** Don't start a PCB design with no idea on manufacturing limitations hoping somebody can manufacture it!



Very different PCB process between an Arduino and an iPhone!

PCB STACKUP FOR THIS CLASS

- PCB:NG will be our Fabrication House for this class as well as our assembly house
- They offer a 4-Layer stackup as shown to the right.
- See https://support.pcb.ng/support/solutions/articles/9000057010-pcb-constraints for PCB:NG Constraints

4-Layer Details

Modified on: Fri, 4 Aug, 2017 at 12:49 PM

Stackup:

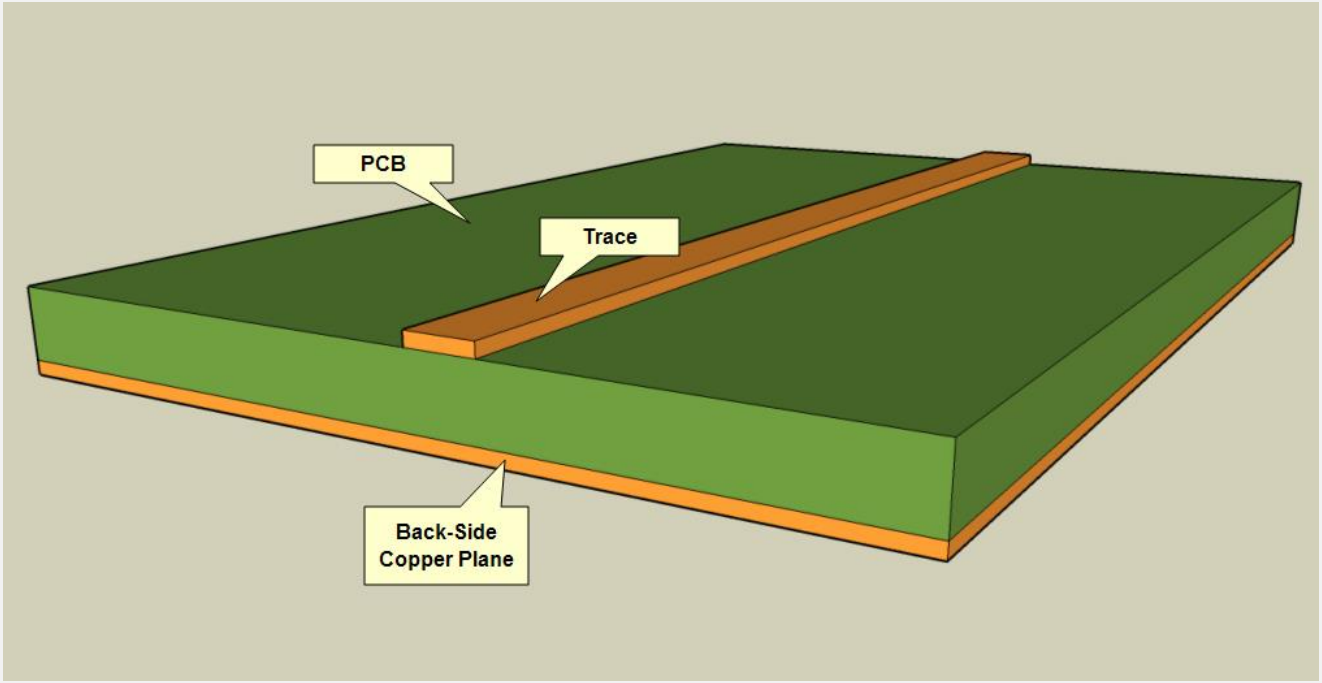
Thickness	Thickness (mm)	Layer	Tolerance	Tolerance (mm)
1 mil	.025	Solder Mask	±0.25mil	±0.01
1.5 mil	.04	1 oz copper	+0.5mil	±0.02
13 mil	.33	FR4 Prepreg		
0.7 mil	.02	0.5 oz copper		
28 mil	.7	FR4 Core		
0.7 mil	.02	0.5 oz copper		
13 mil	.33	FR4 Prepreg		
1.5 mil	.04	1 oz copper	+0.5mil	±0.02
1 mil	.025	Solder Mask	±0.25mil	±0.01

PCB Substrate dielectric constant of 3.95 @ 1GHz

PCB LAYOUT CONSIDERATIONS

MINIMUM TRACE WIDTH

- Refers to the smallest width a copper trace can have.
- Different manufacturers can achieve different levels of trace widths
- **As of 2019:** 4 mils is usually the lowest point without greatly increasing prices.
- PCB:NG -> 4 mil



TRACE WIDTH CONSIDERATIONS

- Tip: Use the biggest trace width you can get away with – but don't overdo it!

- To calculate the trace width for a given current, use a calculator: (https://www.4pcb.com/trace-width-calculator.html)

Inputs:

Current	1	Amps
Thickness	1	oz/ft^2

Optional Inputs:

Temperature Rise	10	Deg C
Ambient Temperature	25	Deg C
Trace Length	1	inch

Results for Internal Layers:

Required Trace Width	30.8	mil
Resistance	0.0164	Ohms
Voltage Drop	0.0164	Volts
Power Loss	0.0164	Watts

Results for External Layers in Air:

Required Trace Width	11.8	mil
Resistance	0.0427	Ohms
Voltage Drop	0.0427	Volts
Power Loss	0.0427	Watts

Inputs:

Current	1	Amps
Thickness	1	oz/ft^2

Optional Inputs:

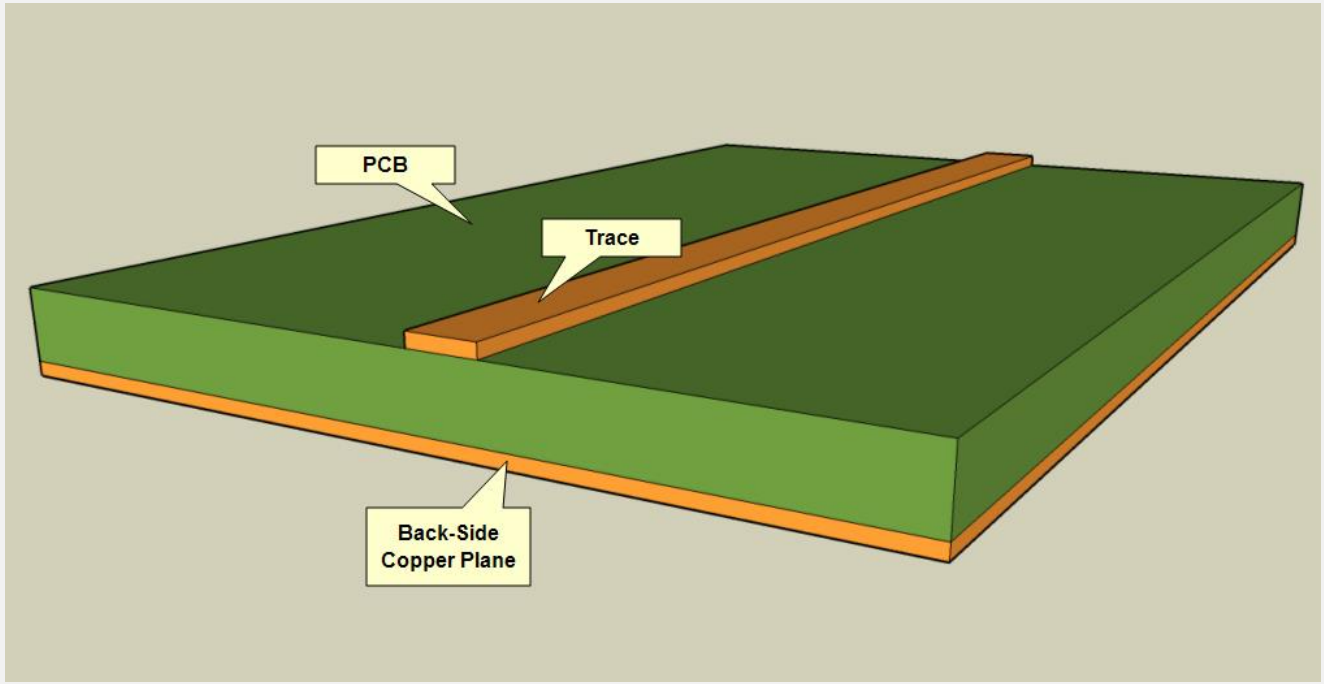
Temperature Rise	10	Deg C
Ambient Temperature	25	Deg C
Trace Length	1	inch

Results for Internal Layers:

Required Trace Width	0.781	mm
Resistance	0.0164	Ohms
Voltage Drop	0.0164	Volts
Power Loss	0.0164	Watts

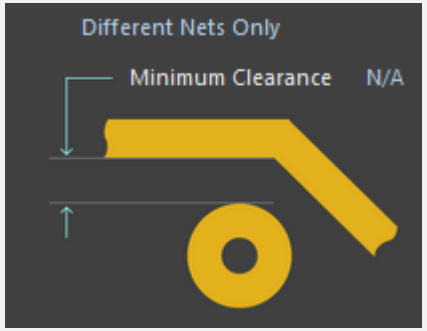
Results for External Layers in Air:

Required Trace Width	0.300	mm
Resistance	0.0427	Ohms
Voltage Drop	0.0427	Volts
Power Loss	0.0427	Watts



CLEARANCE

- Clearance refers to the minimum distance between copper features
- **As of 2019:** 4-mil is the minimum without going to more expensive processes.
- PCB:NG -> 4 mil



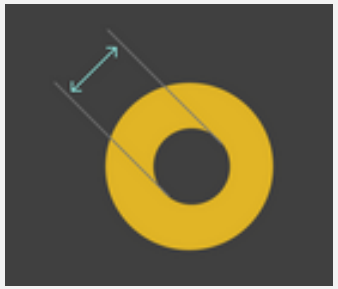
- Best Practice: If you don't need 4 mil clearance, go with something that will make the manufacturer's life easier – 8 mil is usually a good starting point.

- For High voltages: You will want to keep distance from nets that carry high voltages – will not happen in this class, but keep in mind!

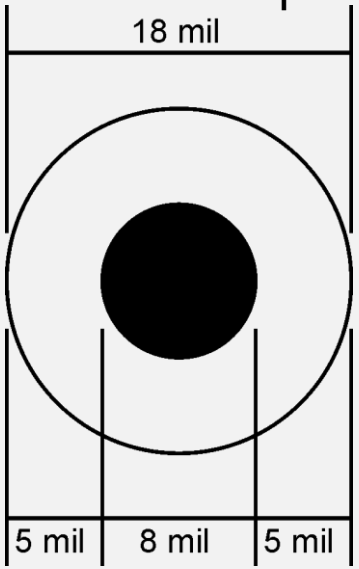
IPC Recommended Electrical Clearance

Voltage	Coated Board	Uncoated(Up to 10,000ft)	Uncoated(over 10,000ft)
0-50	0.13mm	0.64mm	0.64mm
51-100	0.13mm	0.64mm	1.50mm
101-150	0.40mm	0.64mm	3.18mm
151-250	0.40mm	1.27mm	3.18mm
251-500	0.75mm	2.54mm	12.7mm
>500	0.00305mm/V	0.005mm/V	0.0254mm/V

VIA HOLE SIZE

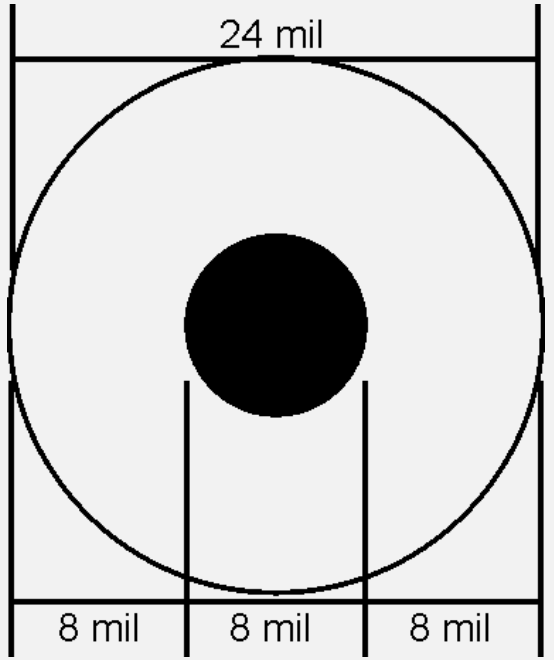


- Drilled vias have two components – the dilled via hole and the Annular Ring, a ring of copper covering the drilled via.
- **As of 2019:** 8 mil is the minimum without going to more expensive processes (laser vias, microvias)
- PCB:NG -> 8 mil holes with a 5 mil annular ring for vias. For mechanical holes or Thru hole components, a 8 mil drill with 8 mill annular ring is the minimum.



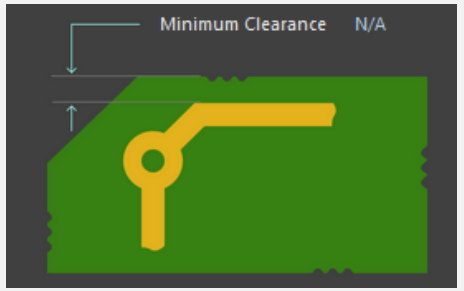
Minimum PCB:NG Via

Minimum PCB:NG Via



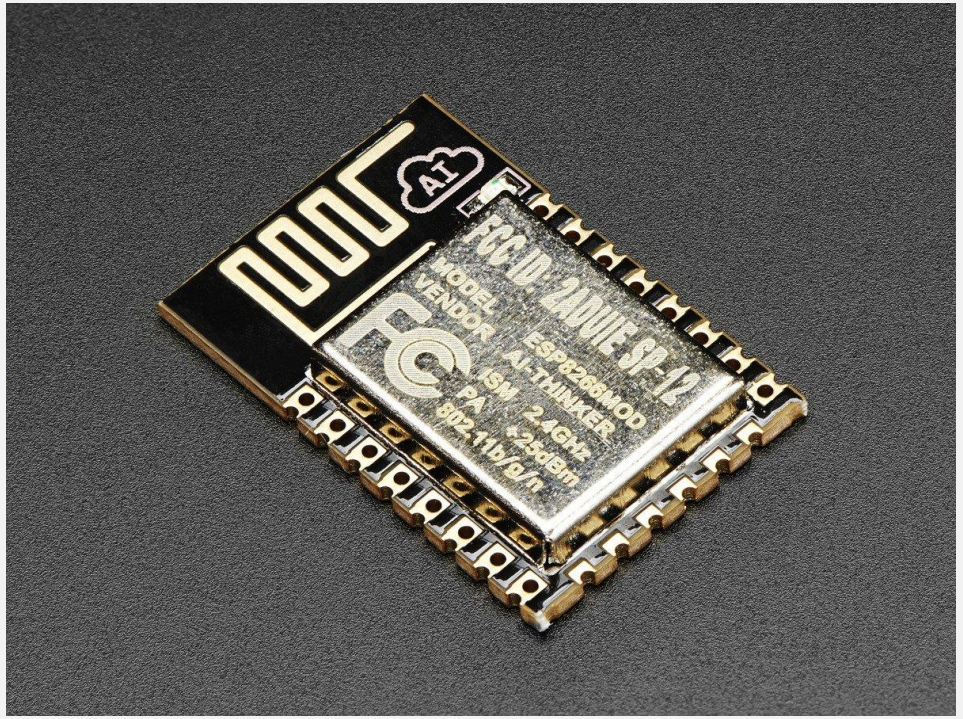
Minimum PCB:NG Hole

CLEARANCE FROM EDGE OF BOARD

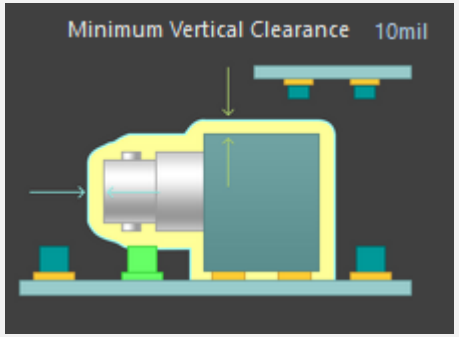


- Refers to the minimum distance between copper and the edge of the board. The manufacturers want some distance so that the process of cutting the boards from the panel don't tear down copper and make short-circuits.
- **Some manufacturers can do recommendations for components such as edge-castellations. Not everybody is willing to do it!**

- **PCB-NG:** 10 mils for outer layers, 20 mils for inner layers.



COMPONENT CLEARANCE

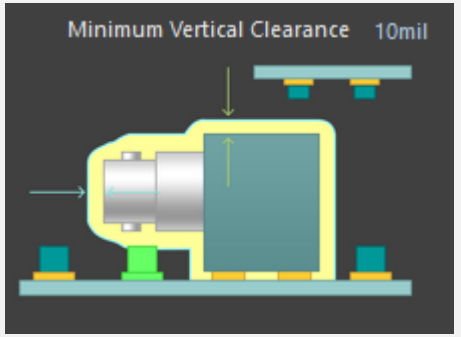


- Components must have some clearance between them – the machine that places the component must be able to reach the component without hitting others!

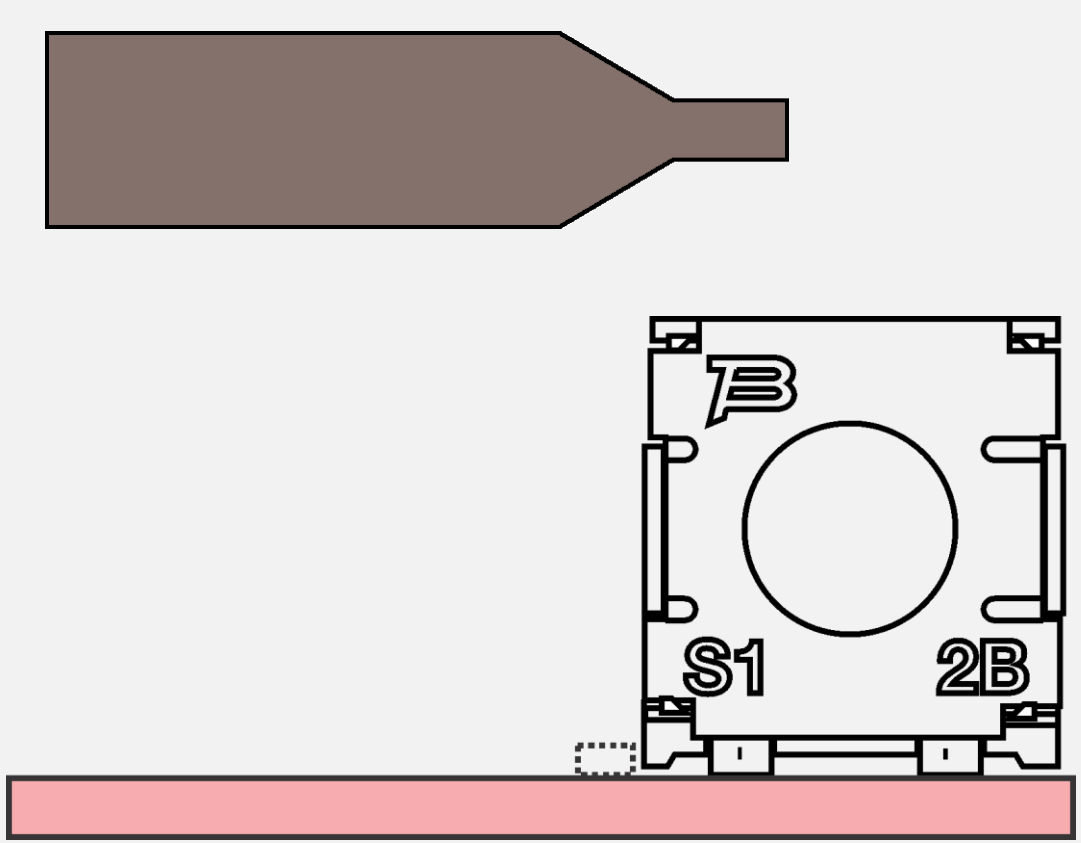
- **Best Practice:** This is where Altium's 3D view shines! It will quickly give you a glimpse of which components may be hard to assemble into a board.
- **PCB-NG** -> Does not specify, but here is a table from Sierra Circuits. I recommend using 10 mils as a complete minimum, but to use as much distance from components as possible without exaggerating.

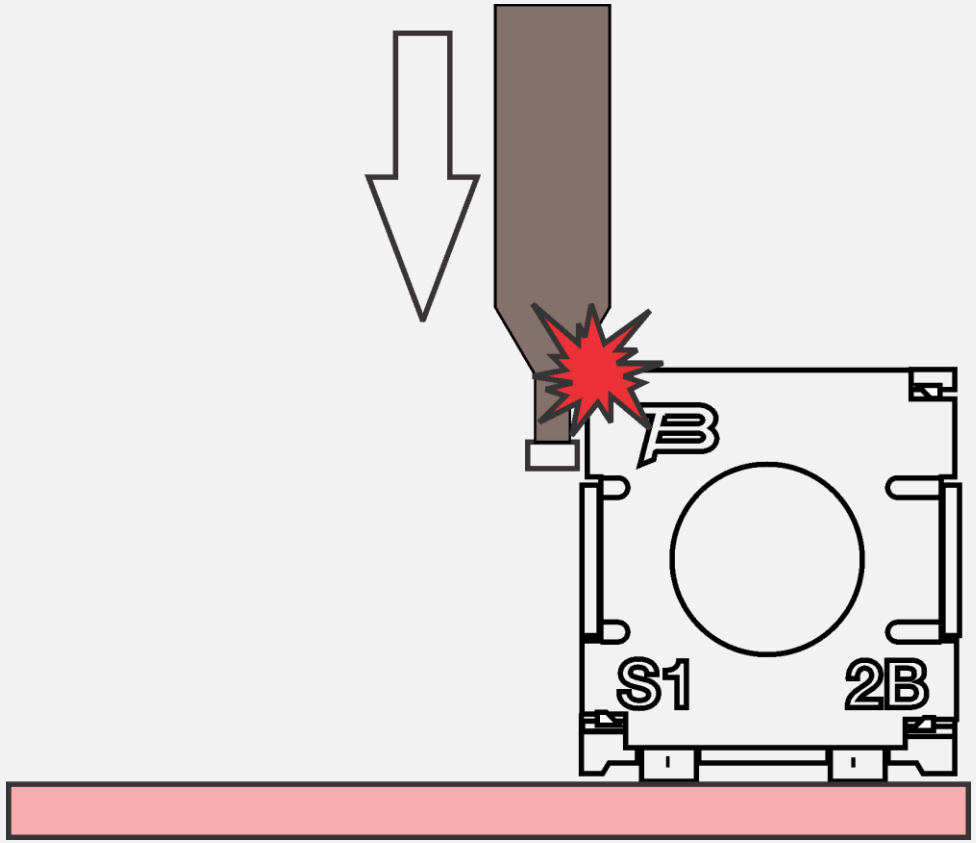
	Chip	SOT	SOIC	PLCC	QFP	BGA	Axial	PTH Conn
Chip	10-20	15-25	20-25	50-75	50-120	75-120	50-75	50-75
SOT	15-25	15-25	20-25	50-75	50-120	75-120	50-75	50-75
SOIC	20-25	20-25	30-50	50-100	75-120	75-120	50-75	100-150
PLCC	50-75	50-75	50-100	100-120	100-120	100-120	100-150	100-150
QFP	50-120	50-120	75-120	100-120	100-120	75-120	100-150	100-150
BGA	75-120	75-120	75-120	100-120	75-120	100-120	100-150	100-150
AXIAL	50-75	50-75	50-75	100-150	100-150	100-150	100-150	100-150
PTH Conn	50-75	50-75	100-150	100-150	100-150	100-150	100-150	100-150

COMPONENT CLEARANCE

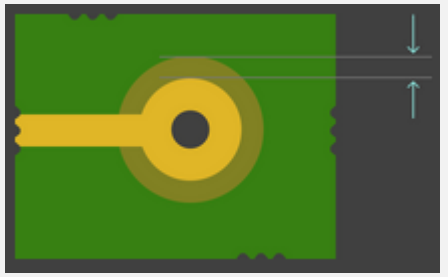


- **Example from PCB:NG**. Be careful with tall components. (https://support.pcb.ng/support/solutions/articles/9000091978-courtyards-and-other-spacing-issues)

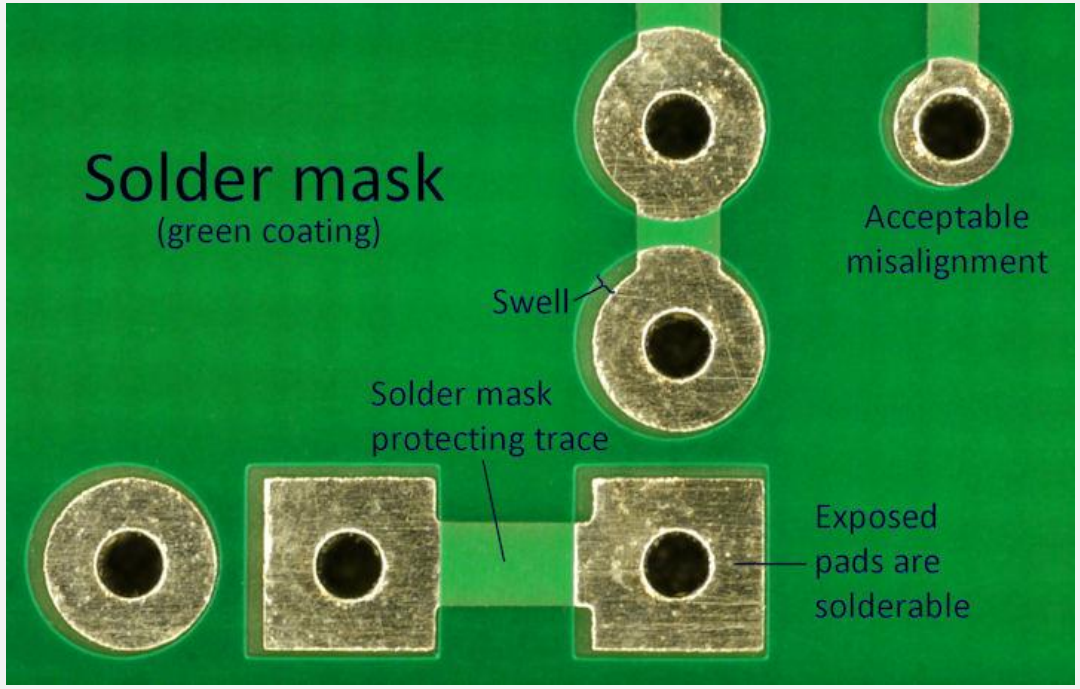


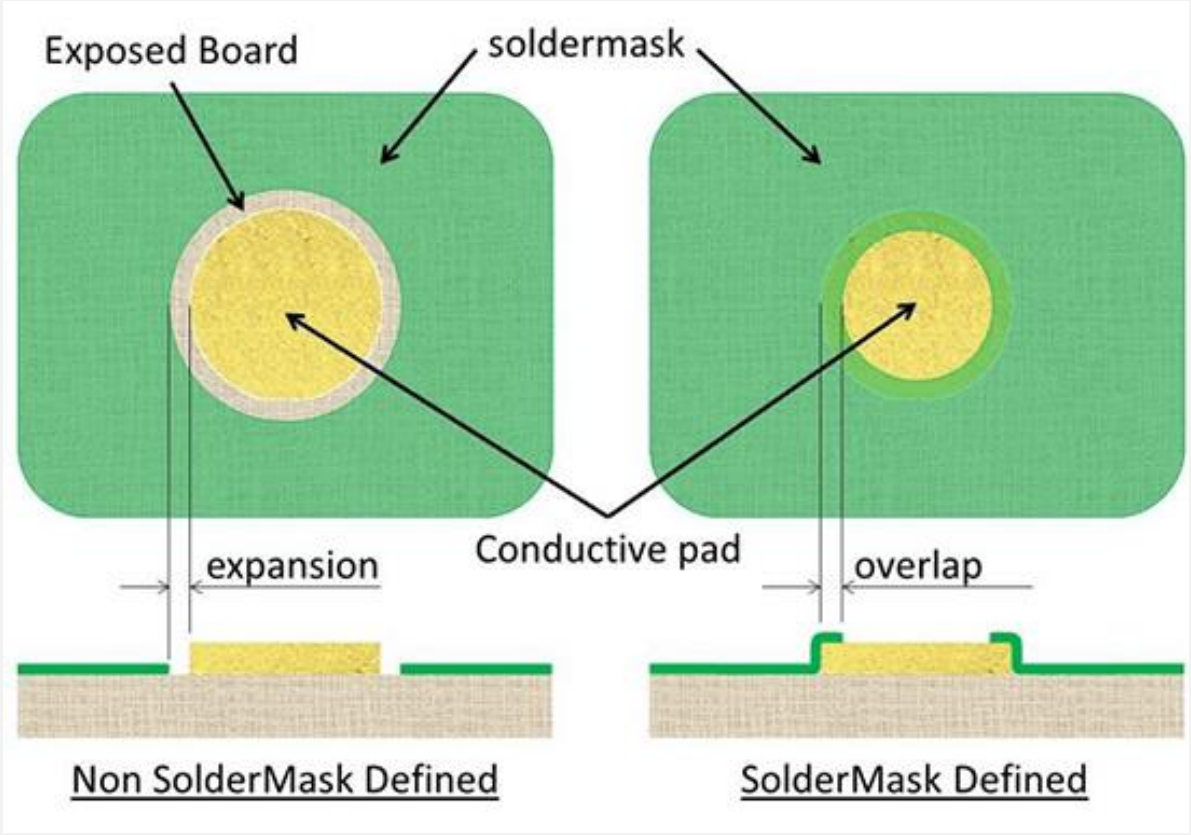


SOLDER MASK EXPANSION

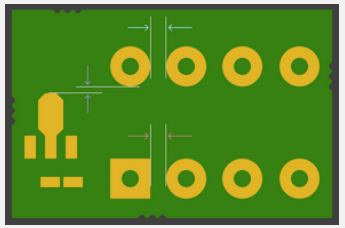


- Refers to opening the solder mask near the pads, to prevent misalignment in manufacturing to result in solder mask covering the copper pads!
- **As of 2019:** 4 mil expansion is common, some places can go to 2mil with an increase in price.
- **PCB:NG ->** 4 mil expansion

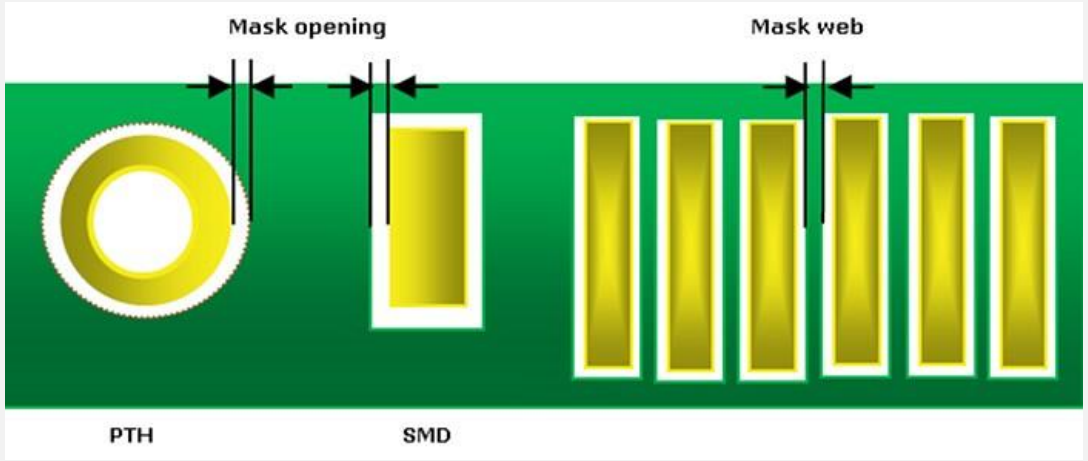




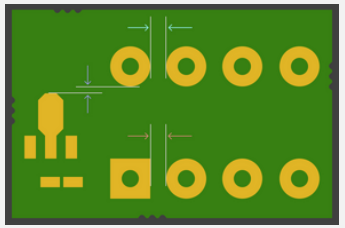
SOLDER MASK SLIVER



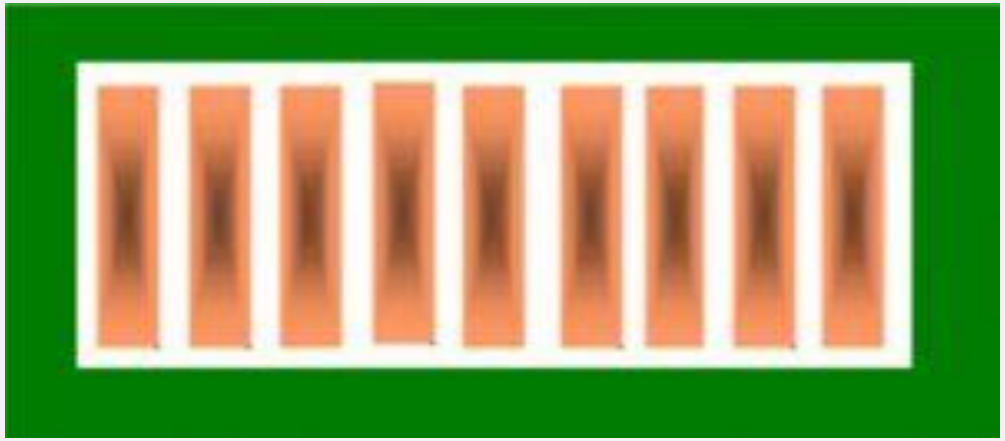
- Refers to the smallest width of soldermask available on a board
- **As of 2019: 4mil** is acceptable in some places, 2mil with other processes
- **PCB:NG-> 4 mil**

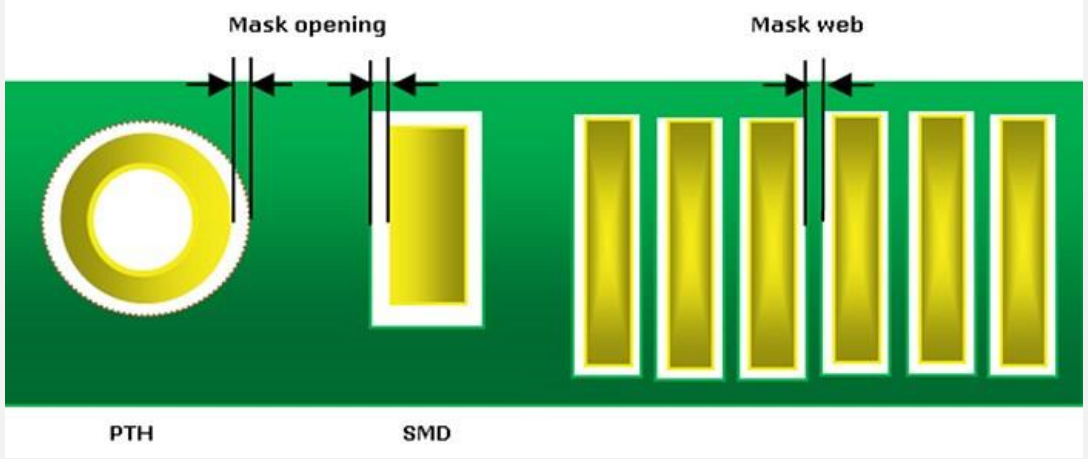


SOLDER MASK SLIVER



- If you cannot achieve 4 mil:
  - See me, you might have a component with a very small pitch
  - Use a gang solder-mask window

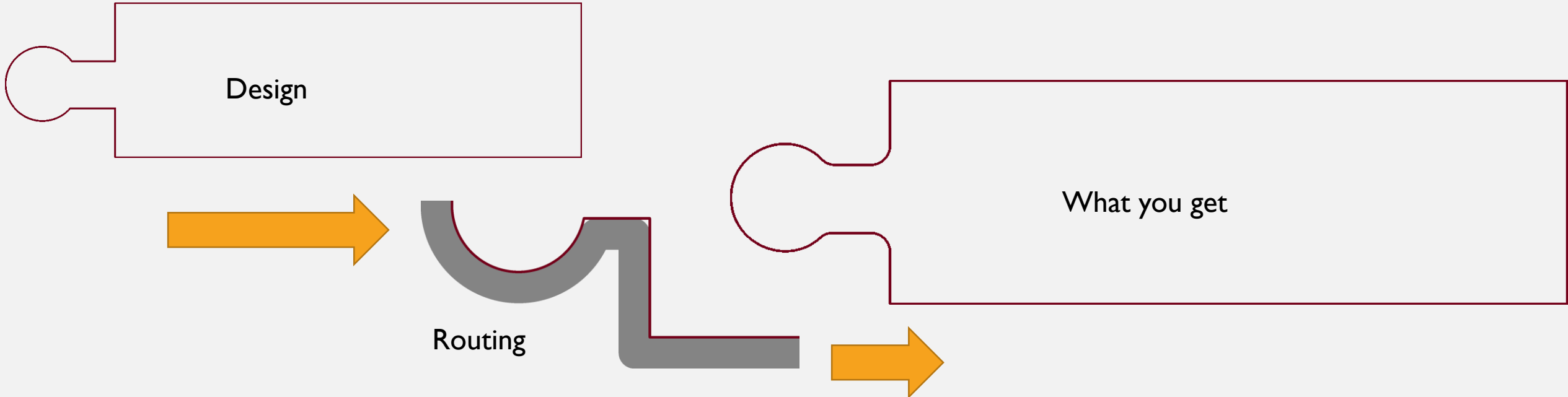




BOARD SHAPES

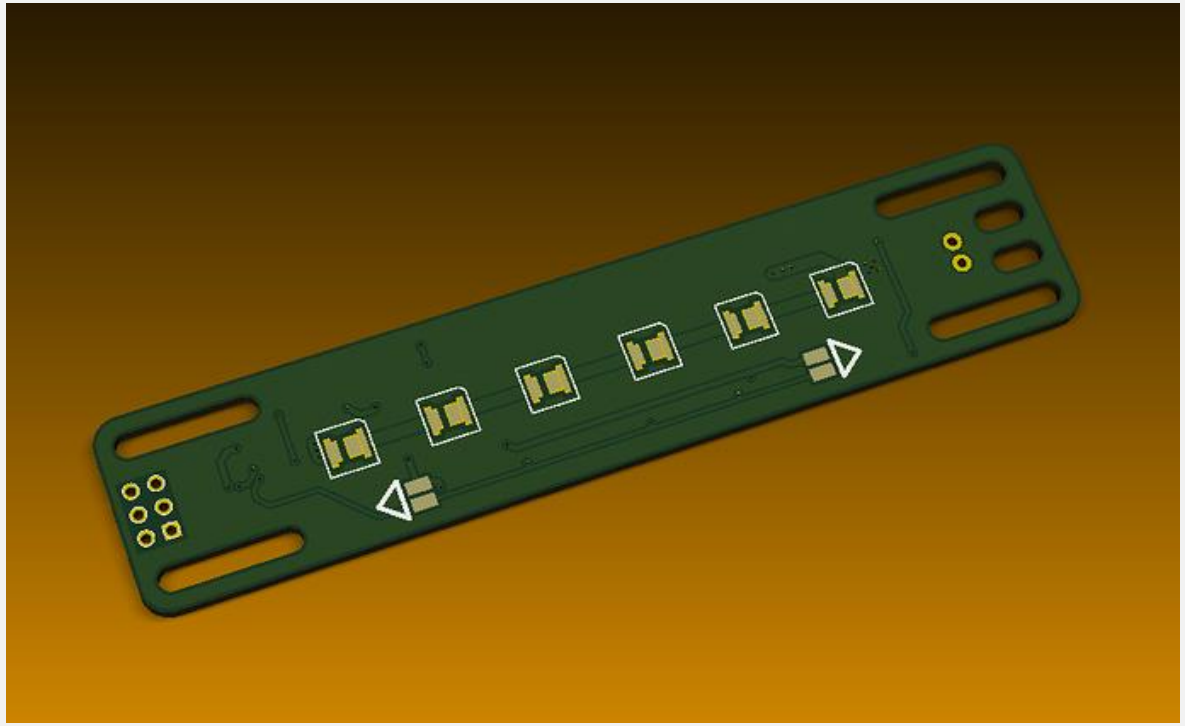
- Boards are either cut (using a special blade) or milled.
  - Cuts (V-cuts) can only do rectangular boards. Cheaper and quicker process.
  - Milled boards are used when a board with a shape different than a rectangle is needed, or when inside slots are needed.

- When a PCB is milled, it will not have a very clear 90* angle on internal corners.



INTERNAL SLOTS AND CUTOUTS

- For PCB:NG: Cutouts are fine, slots are not supported (slots have copper, cutouts don't).



Cutout

Slots

FURTHER READING ON PCB:NG

- PCB Constraints: https://support.pcb.ng/support/solutions/articles/9000057010-pcb-constraints
- Courtyard issues: https://support.pcb.ng/support/solutions/articles/9000091978-courtyards-and-other-spacing-issues
- PCB:NG General specifications: https://support.pcb.ng/support/solutions/articles/9000057005-general-pcb-specifications

FOR NEXT CLASS

- Next class we will do a PCB tutorial for which having a general glance on PCB manufacturing is ideal.

- **Please watch**: https://www.youtube.com/watch?v=_GVk_hEMjzs&t=604s
  - Note: I personally use PCBWay for prototype PCBs and can recommend them.
- **Optional**, if you like EEVBlog: https://www.youtube.com/watch?v=rEB0pl8a5C0
  - Dave from EEVBlog adds some interesting info

- Please read (short, I promise) On text:
  - https://www.palpilot.com/pcb-101/